

Keysight EEsof EDA

Microwave Amplifier Design and Smith Chart Utility for Z Matching Network

Demo Guide



Unlocking Measurement Insights

Introduction – Microwave Amplifier Design

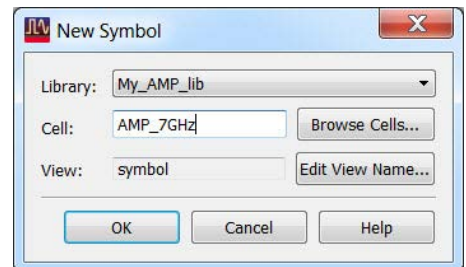
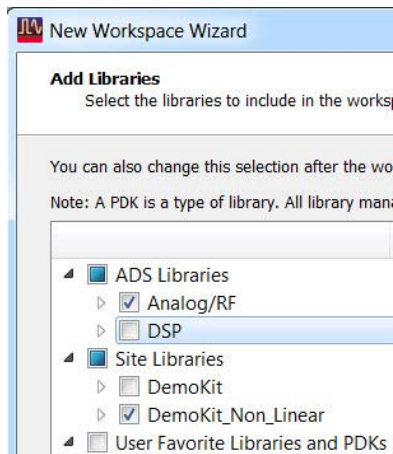
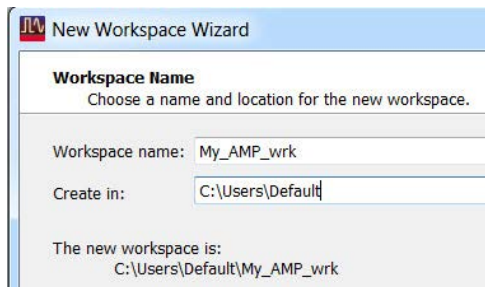
The purpose of this lab is to provide designers with a simple method of developing and designing amplifiers. There are many textbooks available on amplifier theory and design, but they often leave a gap between theory and practical considerations necessary for the production of good amplifiers. A good circuit will compare well with simulated data so that minimal post-production fine tuning is required. This lab combines basic amplifier design theory with practical procedures that are needed to make the design right the first time, minimizing time and effort. Amplifiers are an integral part of any communication system. The purpose of having an amplifier in a system is to boost the signal to the desired level. They also help in keeping the signal well above any present noise so that it can be analyzed easily and accurately. The choice of amplifier topology is dependent upon the individual system requirements and can be designed for low frequency applications, medium to high frequency applications, mm-wave applications, and many more. Depending upon the system in which they are used, amplifiers can adopt many design topologies and can be used at different stages of the system. They are classified as Low Noise Amplifiers, Medium Power Amplifiers, and Power Amplifiers. The most common structure tends to be a Hybrid MIC amplifier. The main design concepts for amplifiers, regardless of frequency and system, remain the same and need to be understood very clearly by designers. Specific frequency ranges pose their own unique design challenges and must be handled appropriately. This paper focuses on the design of small signal C- band Hybrid MIC Amplifiers. This methodology is equally applicable for other amplifiers operating in other frequency ranges, requiring only minor changes to the design procedure.

1. Create Amplifier Workspace

- a. In the main ADS window, click on New Workspace, and when the wizard opens, name your new workspace: My_AMP.



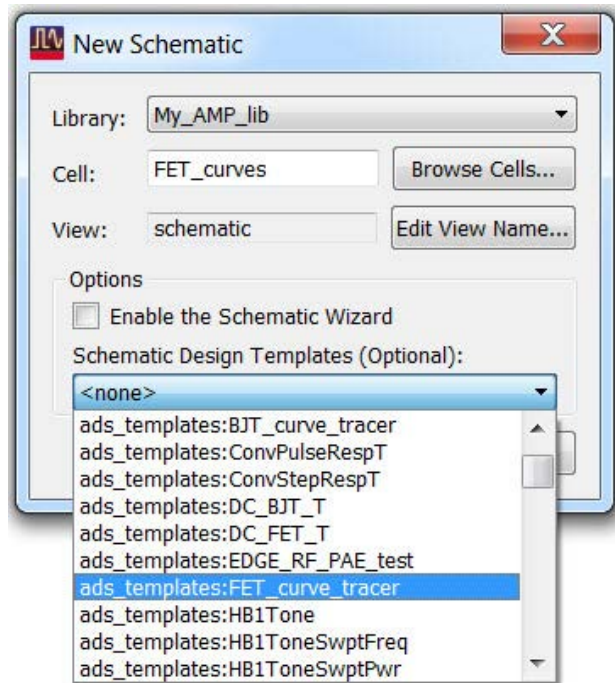
- b. Also name the library My_AMP_lib, and select only the Analog/RF library and the DemoKit_NonLinear library as shown here with the DemoKit technology selected also. Click Next, as needed, and Finish when done.



2. DC - FET Curve Tracer Template

This step uses a built-in DC simulation template that sweeps drain-source voltage at different steps of gate- source voltage (also called a nested sweep). The resulting IDS or bias characteristics are plotted automatically using a built-in data display template. This FET will be used for the amplifier.

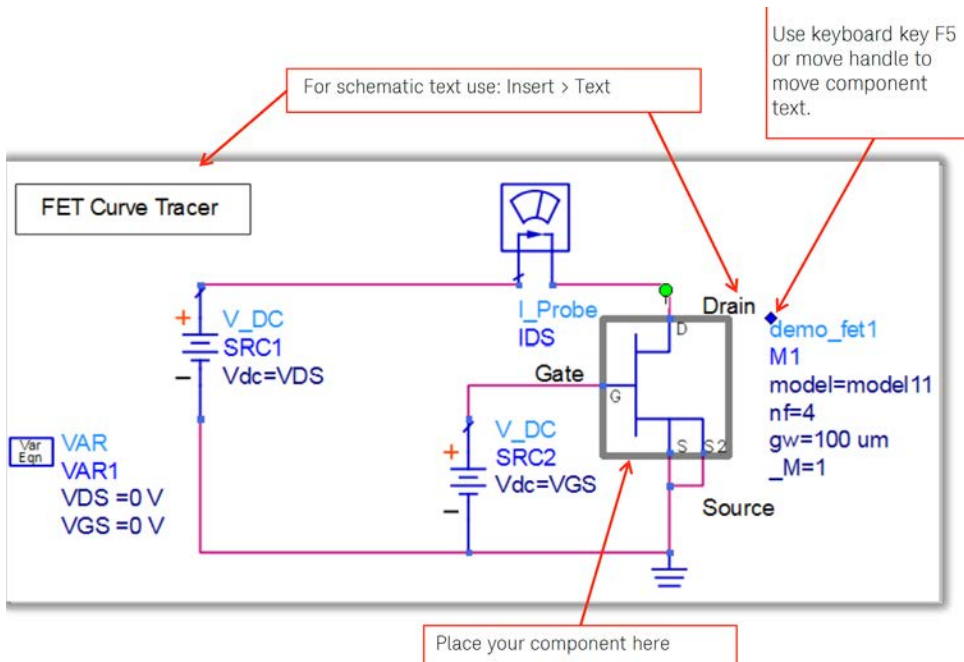
- a. Create a new schematic you should know how to do this - name the new cell: FET_curves and select the ads_TemplatesFET_curve-tracer as shown here.



- b. In the new schematic, select the DemoKit_Non_Linear palette and insert the DEMO FET 1 and also insert the DEMO KIT TECH INCLUDE component shown here.



Wire it with both source nodes (S and S2) connected together and grounded.

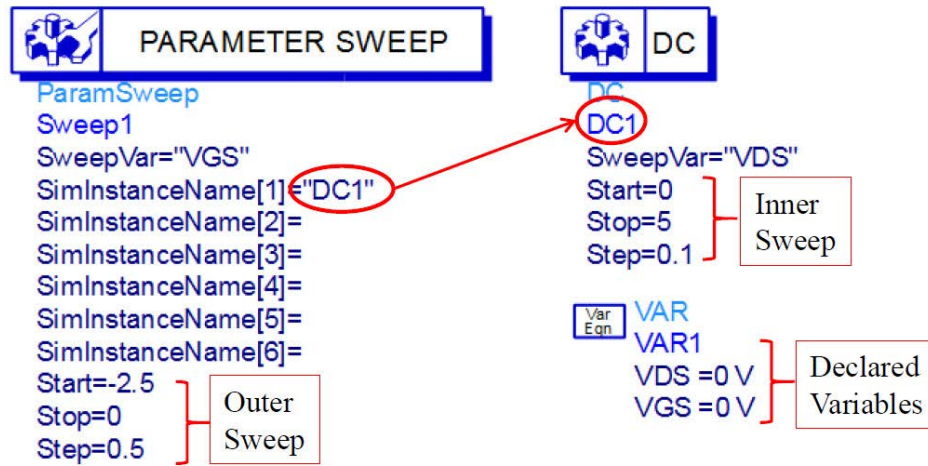


- The FET 1 text may overlap other text, components or wires. Move the component text using the F5 keyboard key or the move handle (blue diamond near component text when component is selected). This keeps schematics easy to read.
- Save the schematic and run the simulation.
- Notice the status window and the data display will pop up immediately after as shown here.

Also, scroll through the status summary and notice that for each value of VGS, VDS is swept from 0 to 5 volts.

| Status / Summary | |
|-----------------------|--|
| CT Sweep1[1] | <My_AMP_lib:FET_curves:schematic> VGS=(-2.5->0) |
| DC Sweep1[1].DC1[1/6] | <My_AMP_lib:FET_curves:schematic> VGS=-2.5 VDS=(0->5) |
| DC Sweep1[1].DC1[2/6] | <My_AMP_lib:FET_curves:schematic> VGS=-2 VDS=(0->5) |
| DC Sweep1[1].DC1[3/6] | <My_AMP_lib:FET_curves:schematic> VGS=-1.5 VDS=(0->5) |
| DC Sweep1[1].DC1[4/6] | <My_AMP_lib:FET_curves:schematic> VGS=-1 VDS=(0->5) |

- Compare the Status information to the Parameter Sweep and DC simulation in schematic to understand how a sweep within a sweep is done in ADS. Notice that the variables must first be declared before they are used. The initial value is not used for the sweep, so they could be 0 volts or 100 volts when declared and the swept values will over-ride them.



- g. In the Data Display plot, the marker value should appear at VDS=3 V and VGS=0.5 V with 75 mA of drain current. Select the marker with your cursor and then use the marker arrow icons to move to different values.

m1
VDS=3.000
IDS.i=0.075
VGS=-0.500

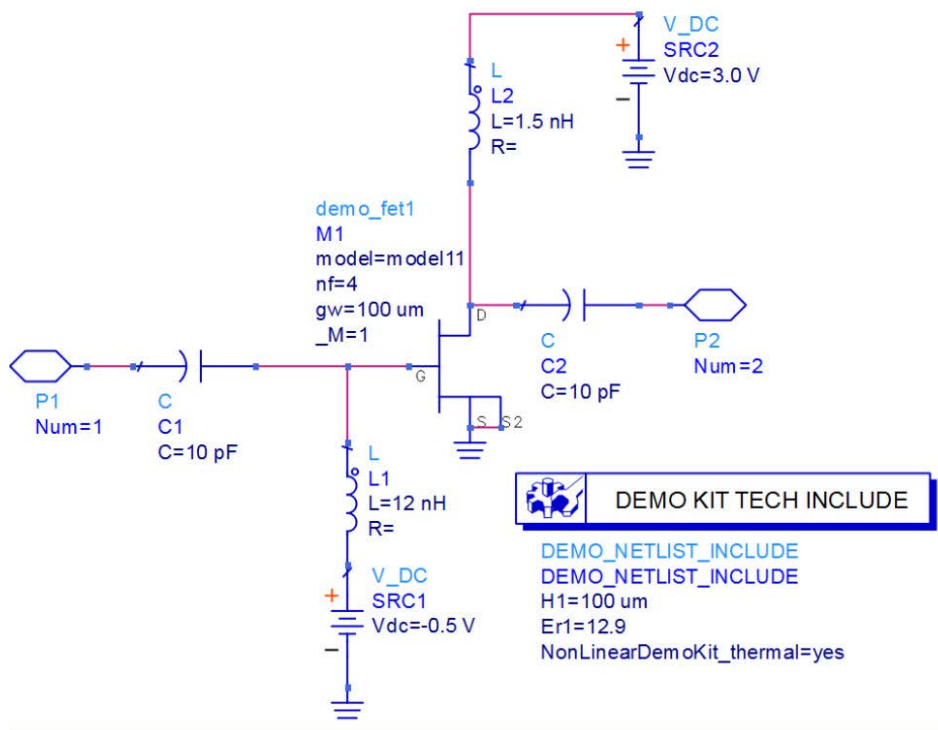
- h. Use your cursor again to select and drag the marker to another trace and try using your keyboard arrow keys to move the marker. Also, try inserting your cursor in the marker readout and type in a different VDS to move the marker.
- i. Move the marker to various traces and values of VDS and notice that the marker (m1) readout and the computed power changes as the marker position changes.
- j. Save and Close the Data Display and Schematic windows. Also, close the Status window.

Now that you know how this FET performs under bias conditions, its time to build an amplifier and test its performance.

3. Build the FET Amplifier Schematic and Symbol

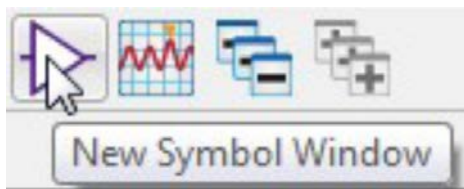
- a. Build the amplifier design shown here – this should take a few minutes following the steps below:
- Create a new schematic and name the cell: AMP_7GHz.
 - Insert the same FET 1 from the DemoKit_Non_Linear palette and insert the DEMOKIT TECH
 - INCLUDE (required for these demo components).
 - Change to the Lumped Components palette and insert L and C as shown – you can edit the L and C values directly on screen or double click to edit.
 - Change to the SourcesFreq Domain palette and insert V_DC sources and set the voltages as shown here: +3 V and –0.5 V.
 - From the toolbar, insert pins on the input and output, grounds and wires as needed.

Your schematic should like the following picture.

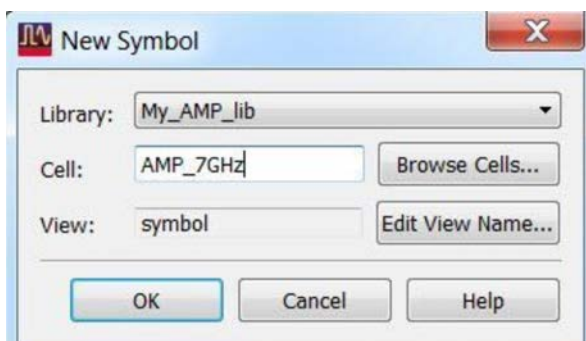


Because ADS supports hierarchical designs a schematic created as a sub-circuit can be placed inside another schematic. The pins were added essentially to establish connectivity for the sub-circuit schematic.

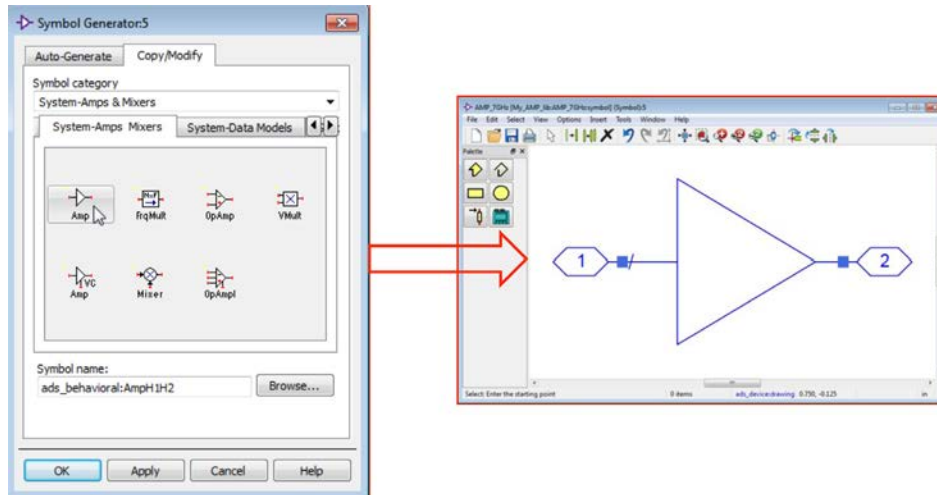
- b. Move component text and set any preference (Options > Preferences) for the display if desired.
- c. Save the design when finished.
- d. In the Main window, click the symbol icon.



- e. In the dialog box, browse to the cell AMP_7GHz as shown here and click OK.



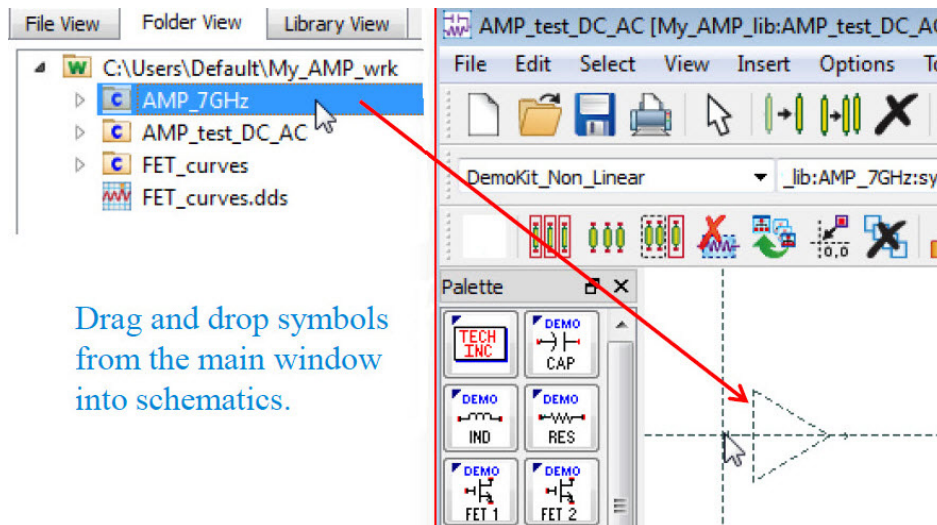
- f. Next, in the Symbol Generator, go to the Copy/Modify tab and find the tab marked System-Amps & Mixers and select the Amp symbol – click OK. Then save and close the symbol window.



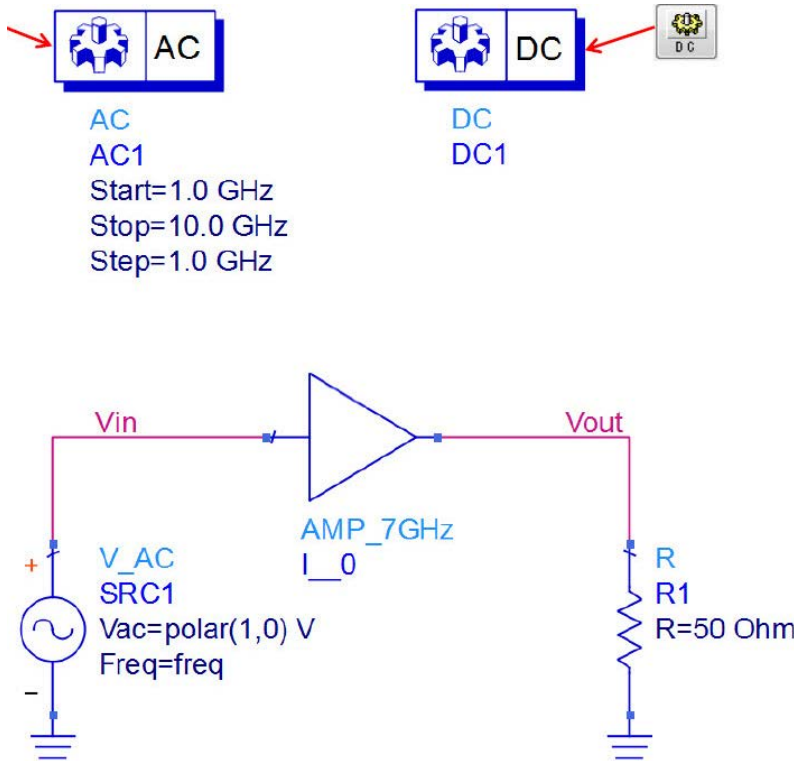
ADS cell AMP_7GHz is now setup to be used on another schematic since the symbol and the ports establish the connectivity to the AMP_7GHz design. In other words this design can be used in a hierarchical design flow.

4. Amplifier DC and AC Simulations

- a. Create a new schematic and name the cell: AMP_test_DC_AC.
- b. In the new schematic, drag & drop the symbol for the amplifier from the Main window.

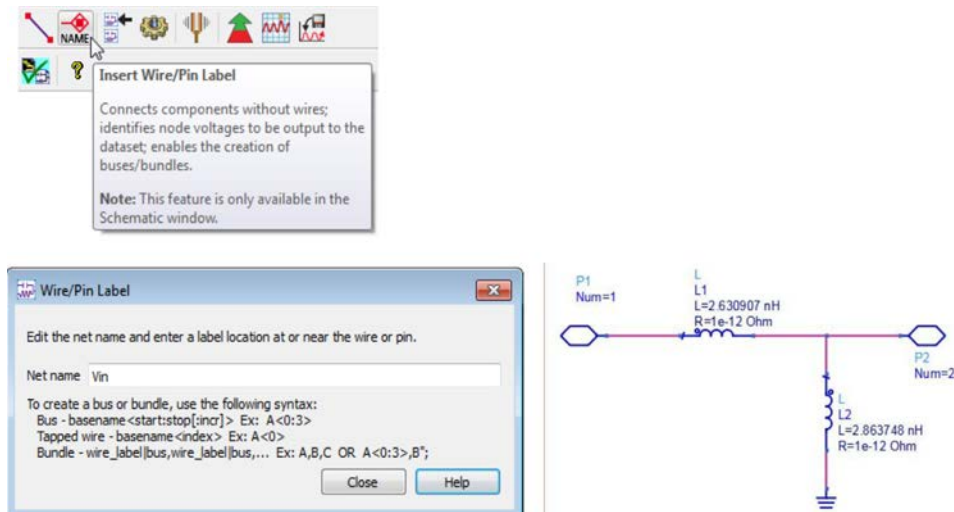


- c. Select the Sources-Freq Domain palette and insert a V_AC source. Add a ground and wire as shown here on the input side of the amplifier.



Also, type in the letter “R” in the component history field, press Enter on your keyboard, and the resistor will be attached to your cursor – insert, ground it and wire on the output side.

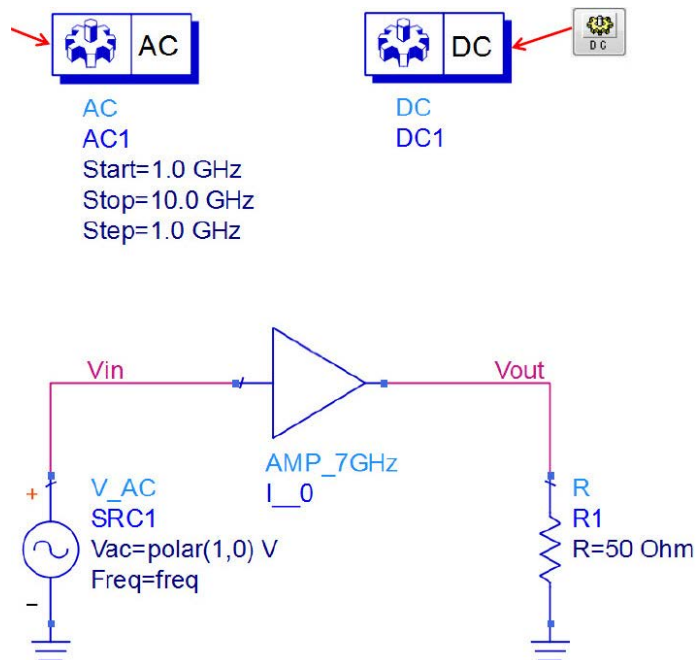
- d. In DC and AC simulation in order to get simulation results you must have either node labels, current probes or power probes in your circuit. Place a wire label by clicking the Name icon (or menu command: Insert Wire/Pin label). Type Vin and select the input wire as shown here – then type Vout and select the output wire. Close the dialog.



5. AC and DC Simulation Setup

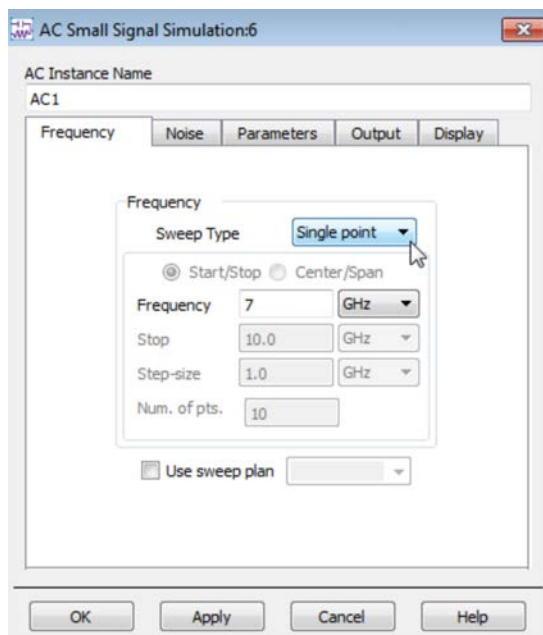
- Select the **Simulation-AC** palette and insert the **AC** simulation controller.
- Select the **Simulation-DC** palette and insert the **DC** simulation controller.

Your design should now look like the one shown here: use the F5 key to move text if necessary. Although the simulation is set-up with a sweep by default, the next step is to change the frequency to exactly 7 GHz and then run the simulation. The DC controller requires no further setup.

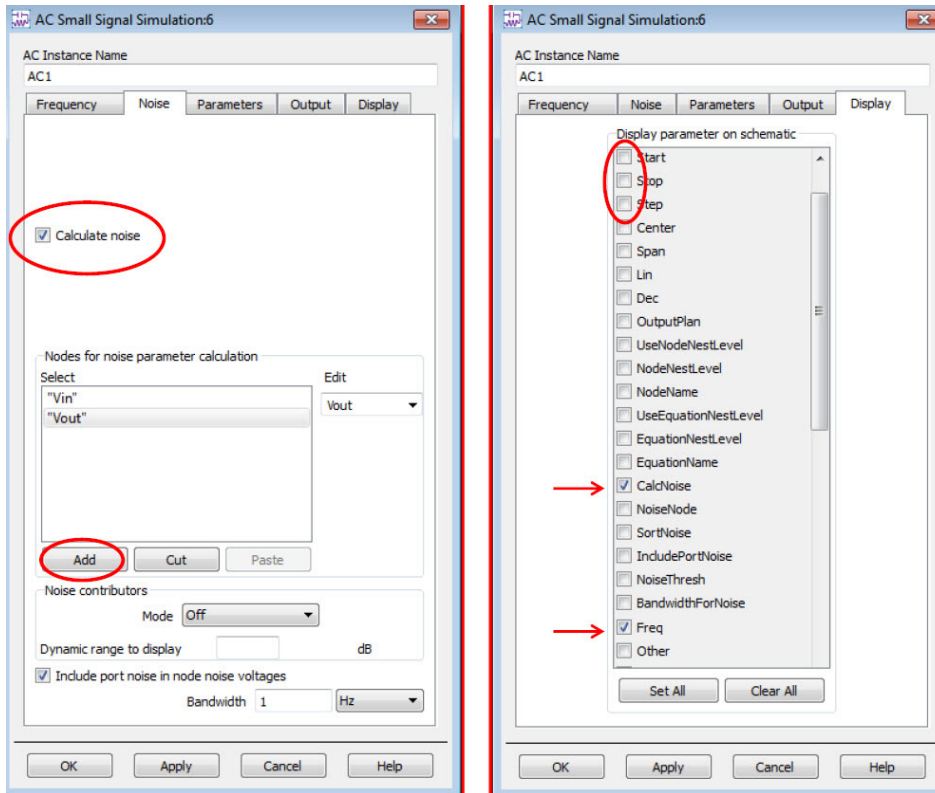


- Edit the AC controller by double clicking on it. In the **Frequency** tab, select the Sweep type to be Single point and set it to 7 GHz as shown here. Click **Apply**.

All ADS simulation controllers have default setup values. But you usually have to specify the settings you want. Also, it is OK to combine different simulation controller on one schematic as long as the sources and circuits are valid for all the controllers, as is the case here with the AC and DC controllers.



- d. Go to the Noise Tab and check the **Calculate noise** box as shown here. Use the **Edit** field to select both **Vin** and **Vout** node labels and click **Add** for each. Click **Apply** –this will give the total noise at the input and output.
- e. Go to the Display tab, scroll down, and turn **off** the Start, Stop, and Step boxes. Then turn on the Freq and CalcNoise. Click Apply. If you always display the parameters you are using, you can edit them on screen. For example, Noise calculations can be easily turned ON or OFF this way.




On your schematic your AC controller should now look like this:

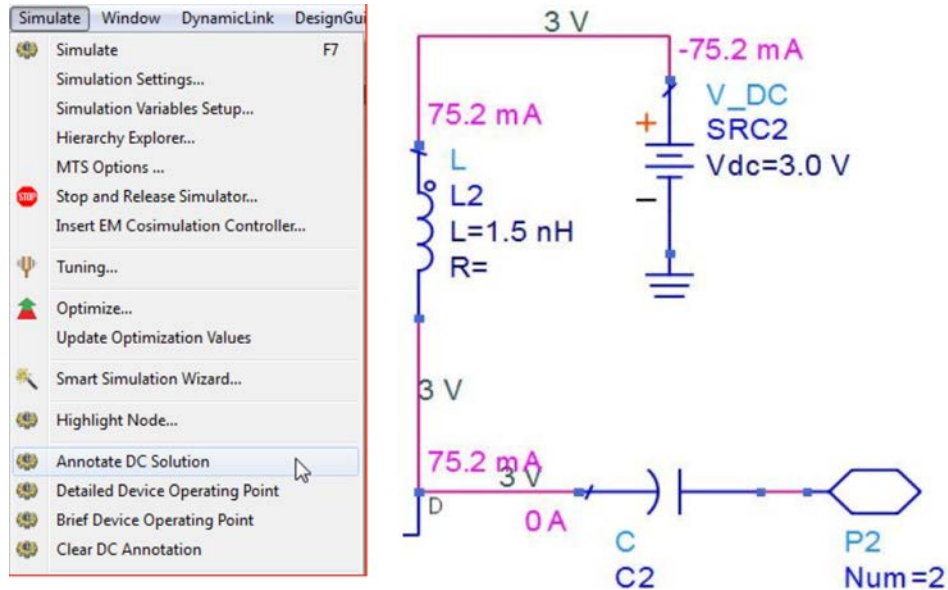


- f. Click OK to close the AC controller dialog when you have the correct settings.

6. DC Annotation in Schematic

Next, you will see the DC node values.


- Click on the simulate icon to run the analyses.
- Once the simulation completes return to the schematic window and Push Into  the AMP_7GHz circuit.
- In schematic, click the command Simulate and scroll down to **Annotate DC Solution**.
- Examine your schematic and you should see the DC bias values (voltage and current) at each node. The minus sign (–) for current is used only to indicate the direction (out or into a node). You should see the values shown here.

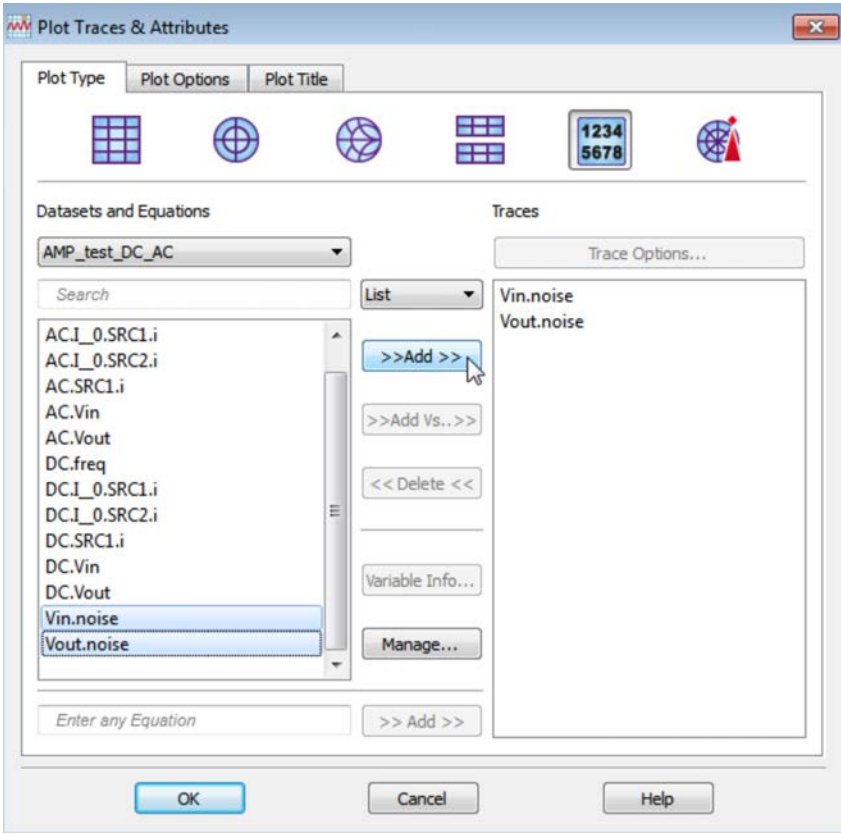


- To clear the annotation, select the command: **Simulate > Clear DC Annotation**. The annotation should clear. Now you know how to get DC values for any circuit quickly: insert a DC controller, simulate, and annotate.

7. AC Data and Noise Results

The next step is to list the noise data.

- In the Data Display, insert a **List** (icon shown here) .
- When the Plot Traces and Attributes dialog appears select both **Vin.noise** and **Vout.noise** as shown here (use Ctrl key) and **Add** them. Click **OK**.



- c. You can change the size of the list by dragging the corner with your cursor as shown. You should have these values:

| freq | Vin.noise | Vout.noise |
|-----------|-----------|------------|
| 7.000 GHz | 0.0000 V | 1.782 nV |

This shows that the noise level at the output pin is about 1.8 nano volts. It will be different at other points in the circuit.

- d. In the same data display insert a column listing. In the Plot Traces and Attributes dialog select both AC.Vin and AC.Vout and **Add** them. Click **OK**.


| freq | AC.Vin | AC.Vout |
|-----------|---------------|------------------|
| 7.000 GHz | 1.000 / 0.000 | 4.486 / -165.113 |

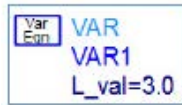
The AC results are in magnitude/phase format by default while the noise results are in volts rms.

8. Variable Equation and Parameter Sweep for Simulation



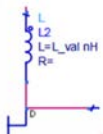
- a. In the schematic, **deactivate** the DC simulation controller; we will not need these results anymore. This is something that you can also do to components if you want them available but not used for a specific simulation.

- b. Insert a **VAR** (Variable Equation) into schematic using the icon shown here. 
- c. Change the name (left side of the equation) to: **L_val**. Use your cursor directly on-screen or edit (double-click) the VAR.



- d. Push into the amplifier subcircuit. Change the drain inductor from 1.5 nH to **L_val n H** as shown here. It is important to keep the units as nH. Otherwise, the value would be in Henrys, which are the default units for inductance. Now, the inductor initial value comes from the variable and has a value of 1.0 nH. But the sweep will vary it. Pop out to the top level schematic.

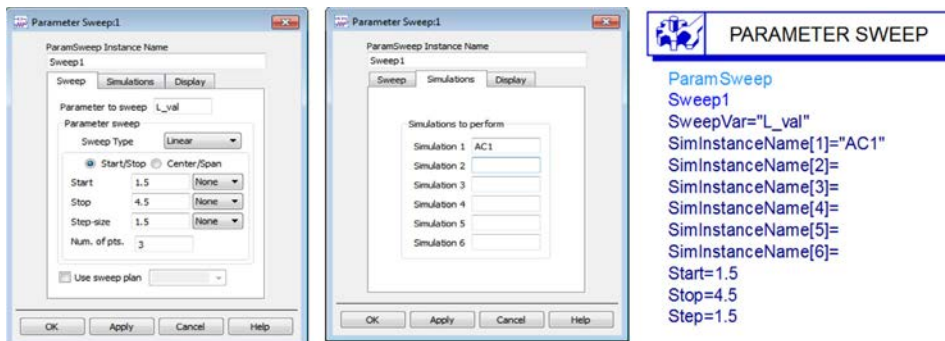
 VAR
VAR1
L_val=1.0



- e. To sweep a variable for AC analysis, you need a Parameter Sweep component, available in any simulation palette. Go to any simulation palette and insert the **Parameter Sweep** as shown here. Next, you need to specify 3 items.



- f. Edit (double click) the **Parameter Sweep** component. In the **Sweep** tab, type in the **Parameter to sweep = L_val**. Then type **Start, Stop** and **Step** values as shown here: 1.5 to 4.5 in 1.5 increments. Next, go to the **Simulations** tab and type the AC controller name (should be **AC1**) and click **OK** when finished.



- g. Notice that the quotes are automatically applied. If you typed in the names on screen, you would have to type quotes. The parameter sweep is now ready.

Run the simulation using either the icon or the F7 keyboard key (default hot key).

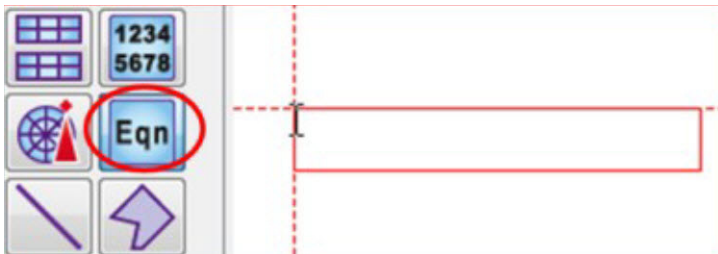
- h. Afterward, look at the data display and notice the changes. Your list now shows the swept values.

| freq | Vin noise | | | Vout noise | | |
|-----------|-------------|-------------|-------------|-------------|-------------|-------------|
| | L_val=1.500 | L_val=3.000 | L_val=4.500 | L_val=1.500 | L_val=3.000 | L_val=4.500 |
| 7.000 GHz | 0.0000 V | 0.0000 V | 0.0000 V | 1.782 nV | 1.869 nV | 1.866 nV |

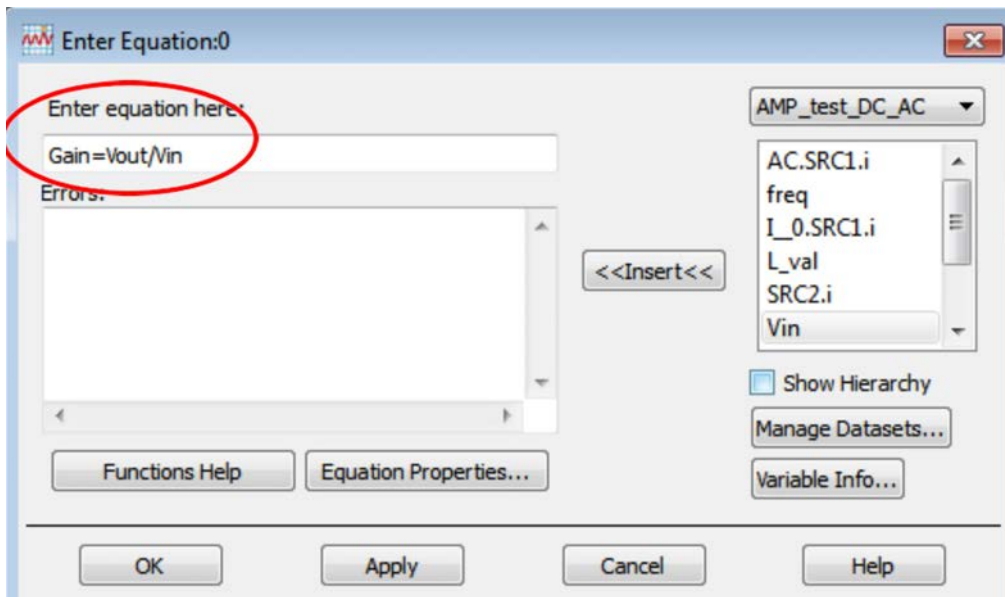
| freq | AC.Vin | | | AC.Vout | | |
|-----------|---------------|---------------|---------------|------------------|-----------------|-----------------|
| | L_val=1.500 | L_val=3.000 | L_val=4.500 | L_val=1.500 | L_val=3.000 | L_val=4.500 |
| 7.000 GHz | 1.000 / 0.000 | 1.000 / 0.000 | 1.000 / 0.000 | 4.486 / -165.113 | 4.693 / 178.950 | 4.681 / 173.469 |

9. Equations in the Data Display

- a. On the data display select the **Eqn** icon and insert an equation as shown here.



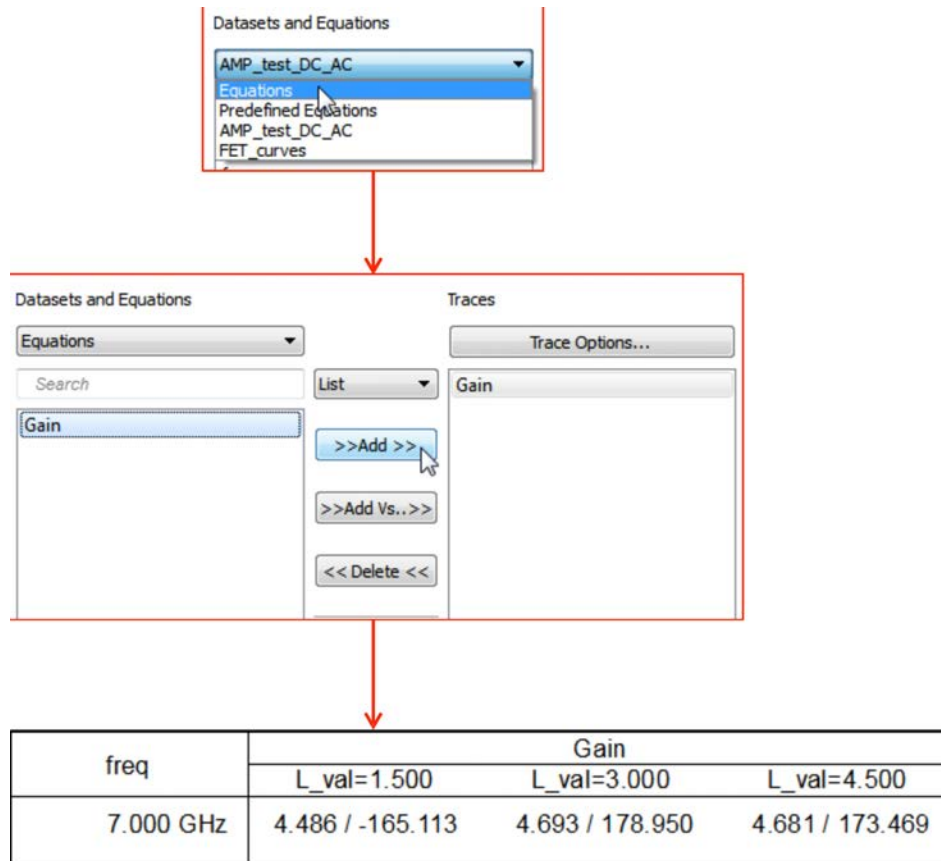
- b. When the Equation editor appears, type in an equation for voltage gain: $GAIN = V_{out}/V_{in}$.



- c. Click **OK** when finished and you will see the equation in black. If it is red in color, it is incorrect.

Eqn Gain=Vout/Vin

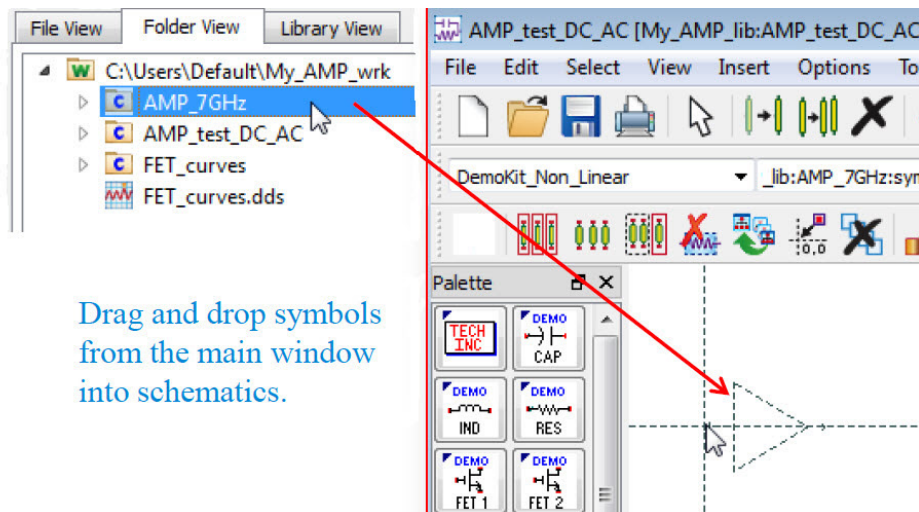
- d. Insert a new **list** (use the List icon). Then select Equations from the drop down list and add the GAIN equation as shown here and click **OK**. Notice the three values of gain for each swept value of L.



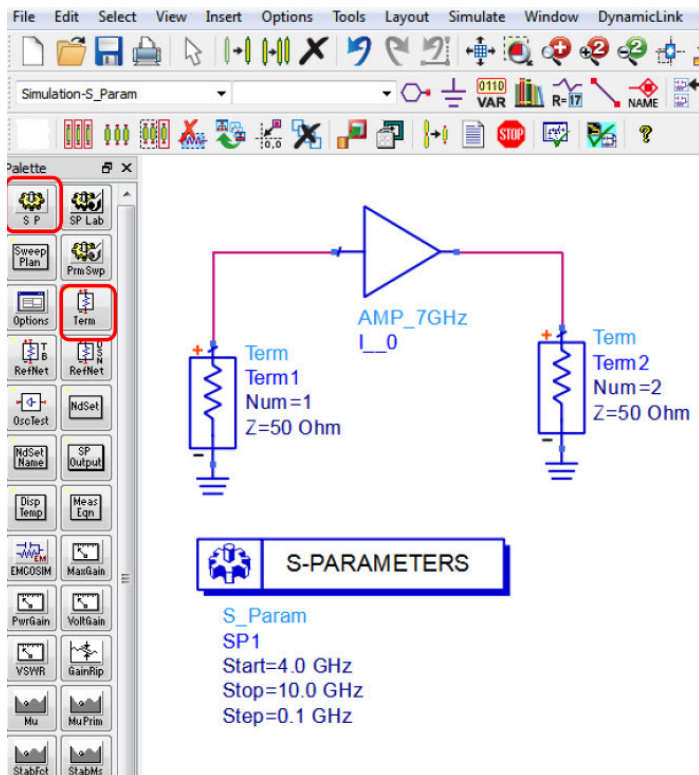
- e. **Save** and **close** the schematic and data display when done.

10. Amplifier Impedance

- From the Main window, create a **new schematic** and name the cell: **SP_amp**. This will be used to determine the input impedance (Z_{in}).
- In the new schematic, **drag & drop** the symbol for the amplifier.



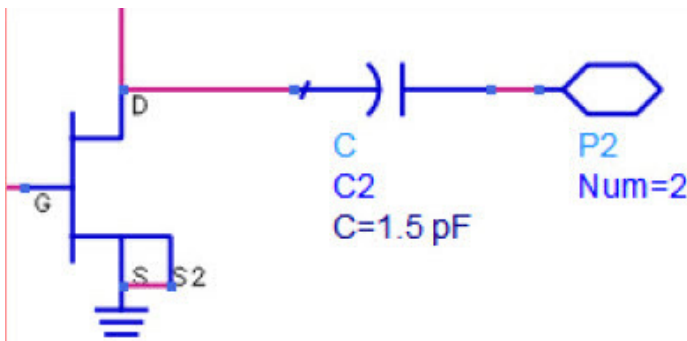
- Then go to the S-parameter simulation palette. Insert the **S_Param** controller, ports 1 & 2 (terminations), grounds and wire as shown here.



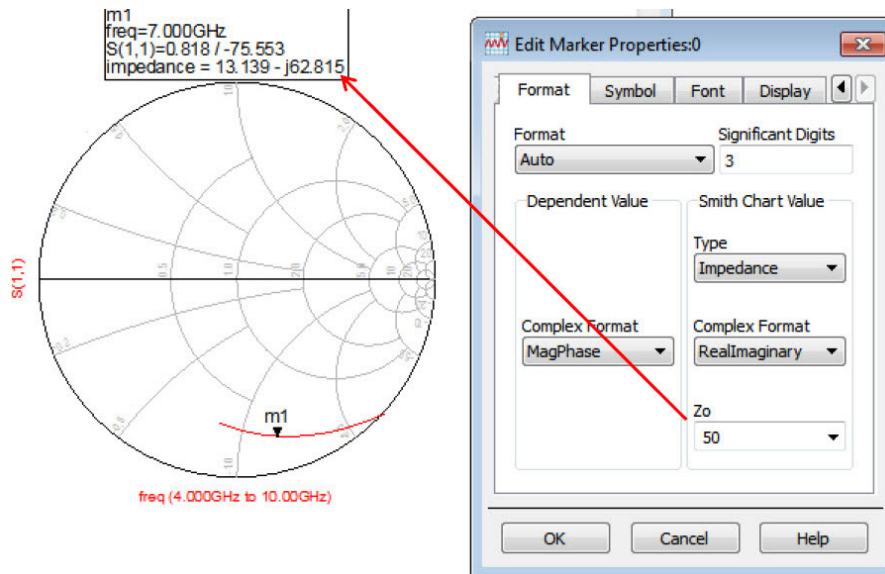
- d. Set the simulation frequency directly on-screen: **Start = 4 GHz, Stop = 10 GHz, Step = 0.1 GHz.**
- e. Instantiate the **L_val** variable block on the schematic and set as shown here.



- f. Push into the **AMP_7GHz** amplifier sub-circuit and change the value of the capacitor connected to the drain as shown below.



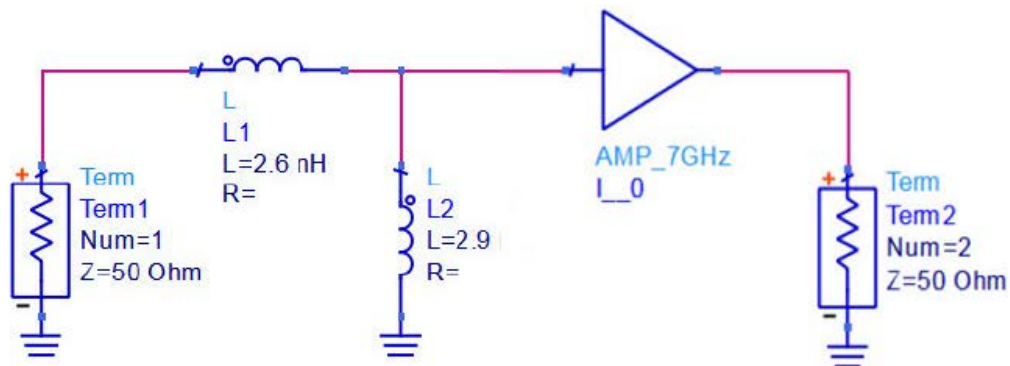
- g. Save the design and run the simulation.
- h. In the new Data Display, insert a **Smith Chart** and add **S (1, 1)** as shown below. Put a marker on the trace at 7 GHz.
- i. **Edit** (double click) the **marker properties** and set **Zo to 50** as shown here. The marker impedance now reads about **13-j*61 ohms**. This value will be used for the match at the amplifier input next. **Save** the data display.



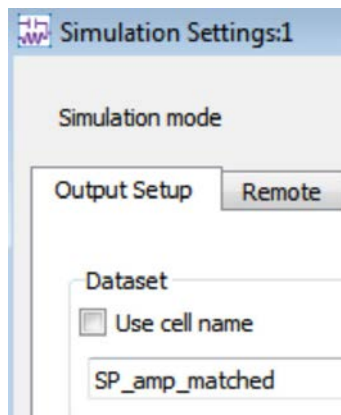
11. Impedance and Gain with a Matching Network

The next step is to add a matching network to improve the results.

- Go back to the **SP_amp** schematic and add a lumped **series** $L = 2.6$ nH, and a **shunt** $L = 2.9$ nH as shown here. This matching network was created using the ADS Smith Chart Utility for impedance matching. It is the optional step at the end of this lab but if you don't have enough time you can try it after this course.

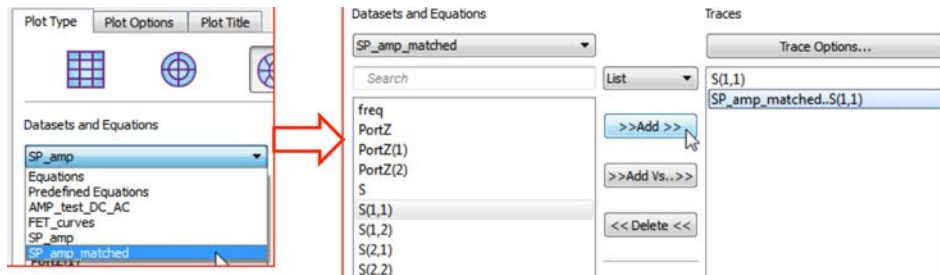


- Use the command: **Simulate > Simulation Settings** and change the dataset name to **SP_amp_matched** and simulate.

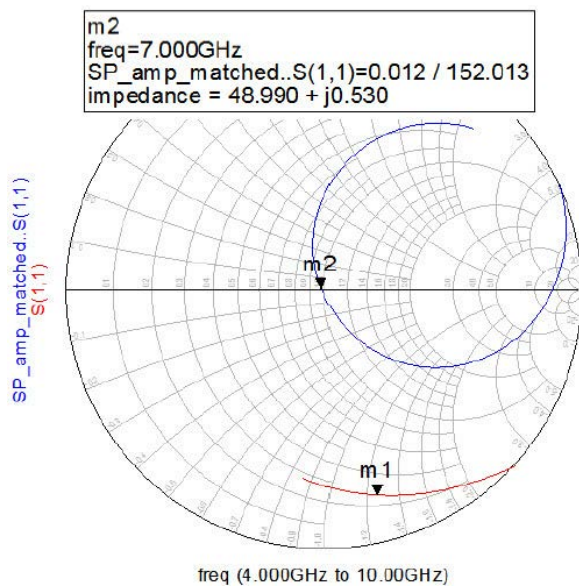


- When the simulation finishes, go to your **SP_amp** Data Display do not change the default dataset (answer No when prompted).

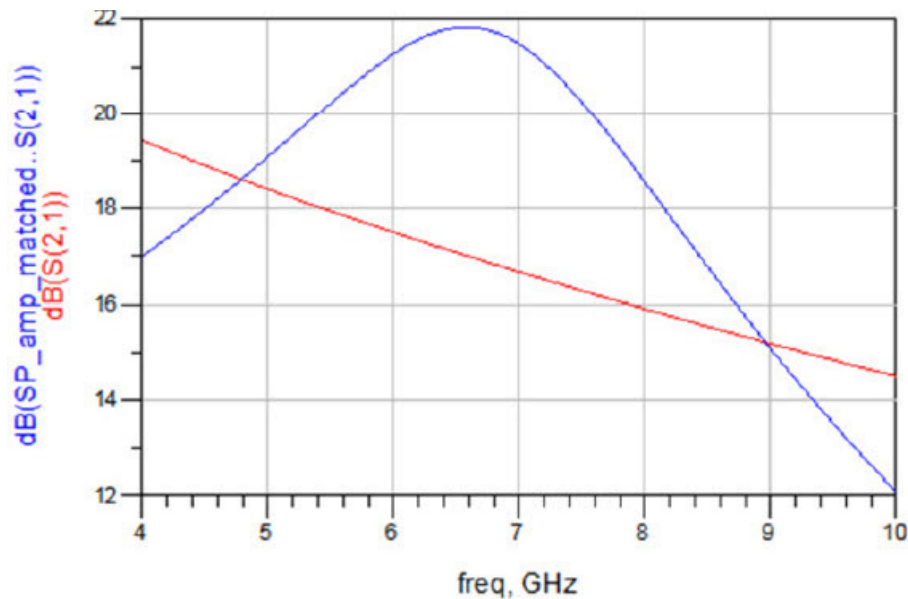
- d. Edit (double-click) the Smith Chart. Select the **SP_amp_matched** dataset and add the **S(1, 1)** trace as shown here.



- e. Place a marker on the new trace at 7 GHz – double click the marker readout and, in the Format to 50 as you did for the other trace. The amplifier input is now matched fairly close to 50 ohms. Of course, the output match, if done, would affect it but for this is only for the lab exercise we are only doing the input.



- f. Insert a rectangular plot and add **S(2, 1)** from the **SP_amp** dataset and then add **S(2, 1)** from the **SP_amp_matched** dataset to see the gain improvement in dB as shown here. The amplifier now has greater than 21 dB of gain at 7 GHz.



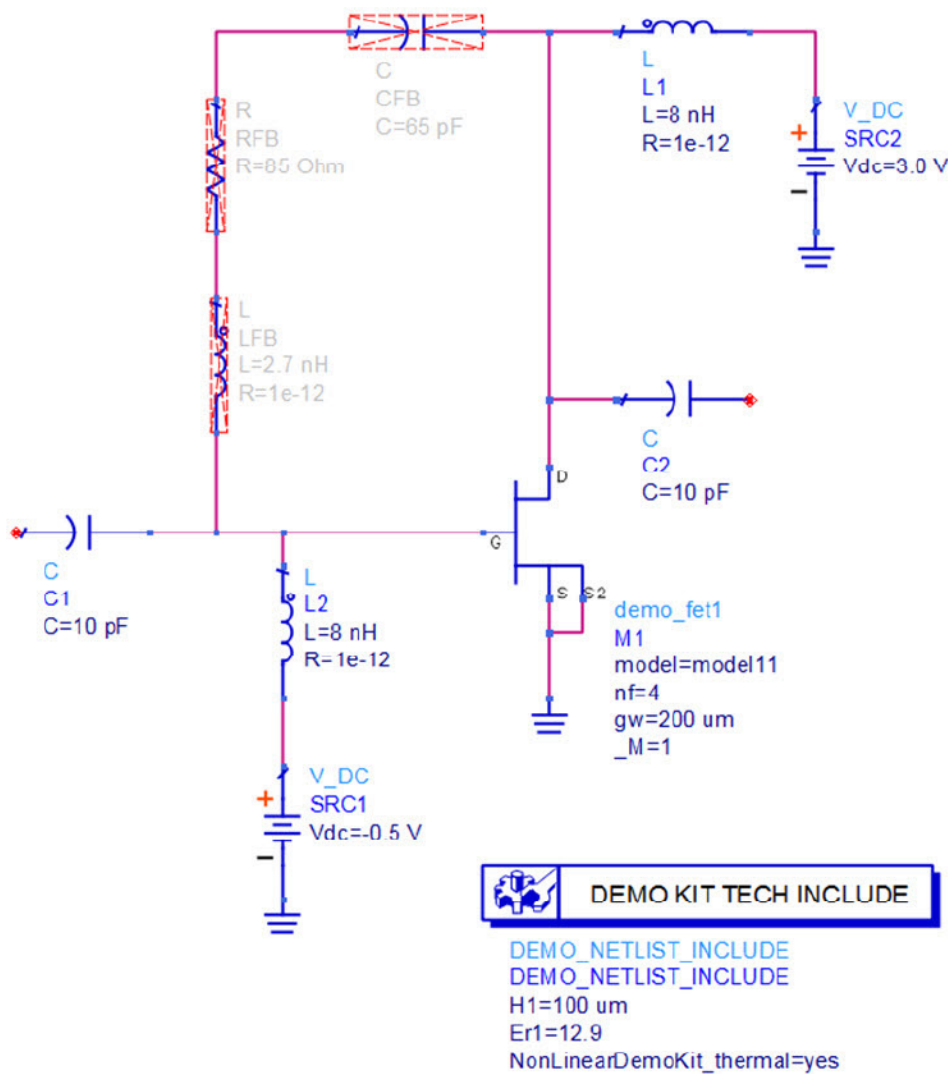
g. Save and Close all windows except the Main window.

IMPORTANT NOTE: Now you have practiced the basics of linear simulation. The optional step at the end of this lab shows how the ADS Impedance Matching tool was used to determine the topology and values for the matching network you just used. It is not necessary to do that step –do it only if you finish early. If not, you can do it at a later date to see how this tool works.

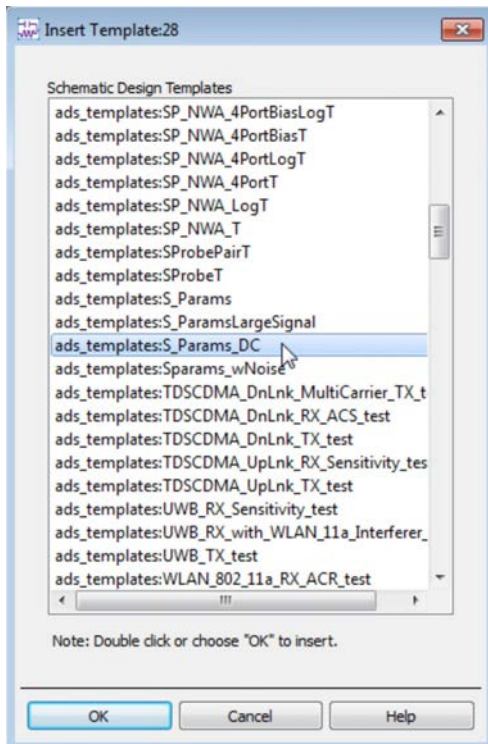
12. Amplifier Stability Design with Template

This part of the lab uses small signal stability analysis. An ADS built in template is used for the stability analysis.

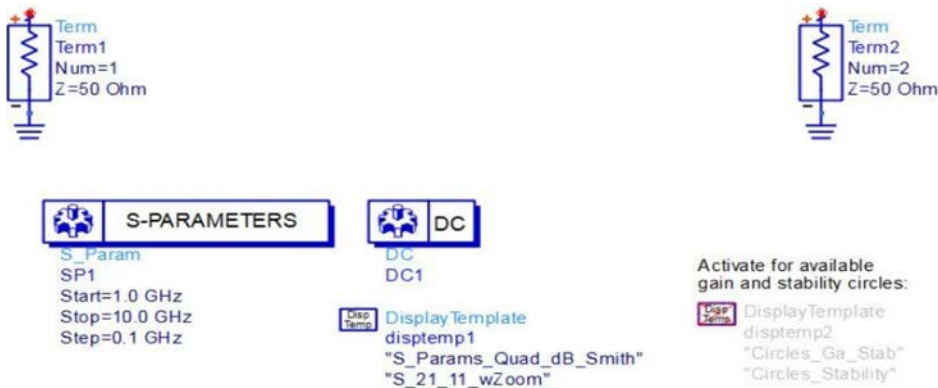
- Open the workspace My_Sys_AMP.wrk that should be located in the c:\Users\Default\ folder.
- Open the schematic view for cell a1_stability.
- Notice that this circuit is a FET amplifier very similar to the one you have been working on.




- d. From the schematic menu select Insert > Template... In the Insert Template window scroll down, select the S_Param_DC template.



- e. Click OK and notice that you will have a ghost image that you can place on your schematic. This is actually a collection of components and controllers, try to place somewhere on your schematic such that it does not overlap the amp design.



- f. Notice that the template has everything you need for the analysis; you just need to provide the design to be connected between the two Terms. Also notice there are some display templates also selected, these will configure the data display for you.
- g. You may move the two Terms if you need to. Connect a wire from Term1 to the open pin of capacitor C1 and Term2 to the open pin of C2.
- h. Run the simulation. Once the data display opens you will notice that the display templates have already configured several plots for you. The data display contains two pages. Observe and experiment with some of the settings by following the instructions provided in the data display.

- i. Select the schematic window and activate  the DisplayTemplate: disptemp2.

Activate for available
gain and stability circles:

Disp
Temp DisplayTemplate
disptemp2
"Circles_Ga_Stab"
"Circles_Stability"

- j. Run the simulation. Notice that in the data display two more pages have been added. At the bottom of the data display page select tab "Page 3: Circles_Ga_Stab".
- k. Move the **RF Frequency Selector** slider to 7 GHz. Notice that the stability factor is less than one, indicating a possibly unstable circuit.

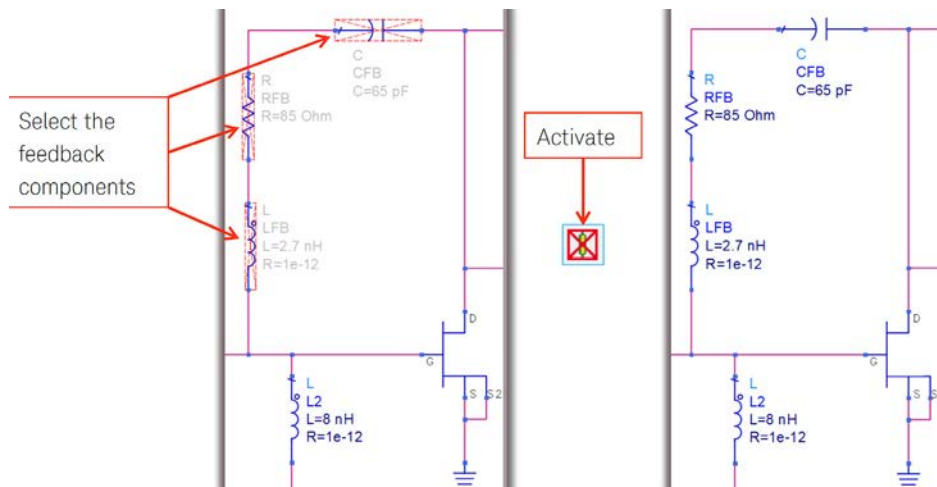
Stability
Factor, K
0.194

- l. On page 4 of the data display after setting the **RF Frequency Selector** slider to 7 GHz you can observe the source and load stability circle on the Smith Chart as well. Also notice the plots for μ_{source} and μ_{load} .

13. Simulate Stable Amplifier

The amplifier is potentially unstable in the next steps you will add the stability circuit components are re-examine the stability results.

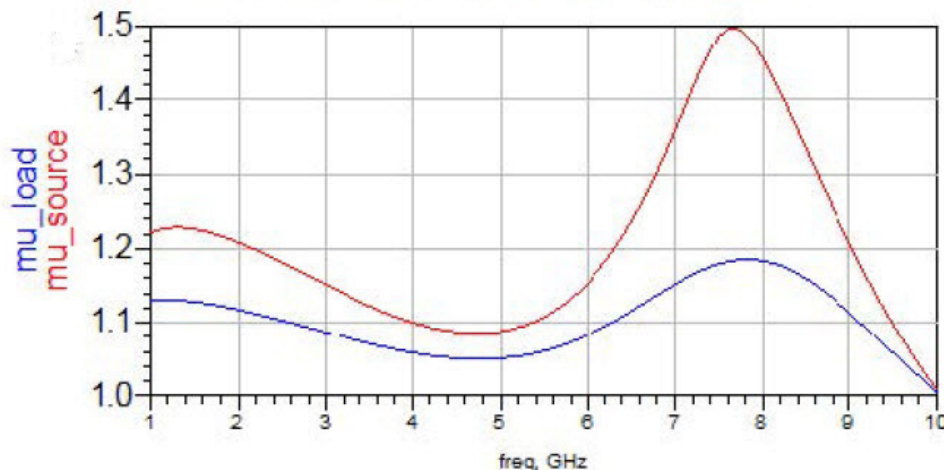
- a. In the schematic window activate the feedback components.



- b. Run the simulation. Notice on page 3 of the data display the circuit stability factor is now greater than 1. On page 4 both μ plots are greater than 1 which is a sufficient condition for an unconditionally stable amplifier.

| | |
|----------------------------|---|
| Stability Factor, K | MaxGain is the maximum available gain if $K > 1$. If $K < 1$, it is the maximum stable gain, $10 \cdot \log(S_{21} / S_{12})$. |
| 1.105 | MaxGain |
| | 15.550 |

If either μ_{source} or μ_{load} is > 1 , the circuit is unconditionally stable.

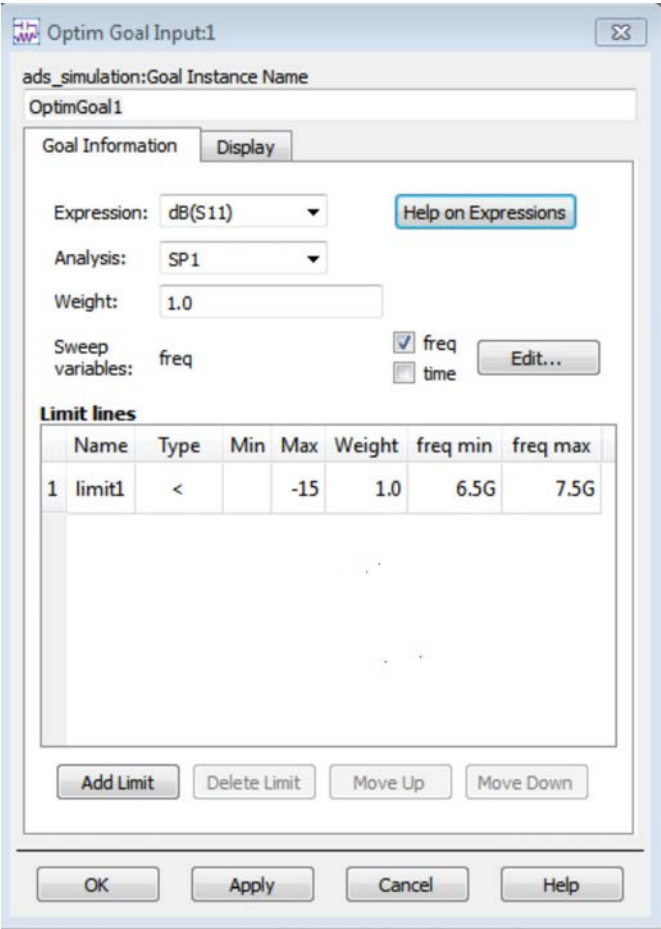


- c. Save and close the schematic and data display window.

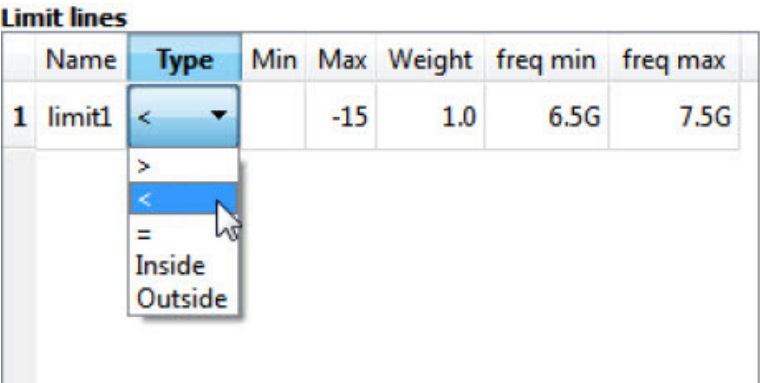
14. Set Up an Optimization Controller and Goals

In the remainder of this lab you will perform an optimization for matching a FET amp to 50 Ohms. This will teach you how the optimizer and goals are used. Because many designers use optimization for many purposes, this lab will teach all the basics.

- Open the schematic view for cell a2_match_opt. This is the same stable amplifier with an added output match circuit formed by L3 and C3. The input match is formed by C1 and L2.
- In this schematic the optimization controller and the goals have been set up for you to save time. These components are located on the Optim/Stat/DOE palette.
- Open the OptimGoal1 setup window by double clicking on it. Notice that in the Expression field **dB (S11)** means that when the simulation is performed it will calculate S11 in dB and this is the measurement used for our goal (requirement or specification). The analysis is set to **SP1** since it will be this analysis that will provide the data for the measurement. The check next to the variable **freq** in the sweep variables adds it to the limit lines so that the goal can be further specified over a frequency range.



- d. In the limit lines the type has been set to < (less than), this tells the optimizer that this goal must be –15 dB (max value) or less for a successful optimization. **freq min & freq max** are set to 6.5 G and 7.5 G. This means that the optimization will only operate from 6.5 GHz to 7.5 GHz. Also, G is used (same as e9) because freq is a reserved variable and uses Hertz by default. Click OK.



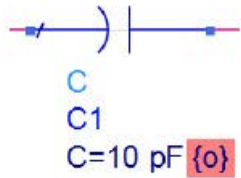
- e. **Optimization controller.** For this lab exercise, the optimizer type has been change to Gradient because it is much more efficient for this problem than Random. **MaxIter** = 50 means that the optimizer will run for up to 50 iterations to achieve the goals. The Normalize goals setting means that all goals will have equal weighting.



NOTE: The ‘Save’ parameters on the Optim controller that are set to ‘no’, means that those values will not be written into the dataset.

15. Enabling Variables or Parameters to be Optimized

- a. **Edit** (double click) the input capacitor (C1). When the dialog appears, click the **Tune/Opt/Stat/DOE Setup** button.
- b. In the **Optimization** tab, set the Optimization Status to **Enabled**. Then type in the continuous range: **0.5 p F to 20 p F**. Click **OK** twice and the component text will show the *opt* function {o} next to the capacitance value.



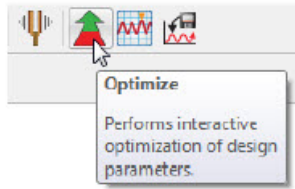
Now you know how to enable and display the values for a single component. However, there is another way to control variables for optimization, especially if multiple variables are to be tuned, optimized, or use for statistical analysis as you will now see.

- c. Click the command **Simulate > Simulation Variables Setup**. Select the **Optimization** tab and notice that your enabled capacitor and values appear. This setup makes it easy to set multiple parameters or variables (VARs).
- d. Set **capacitor C3** and **two inductors** (L2 and L3) to be optimized (check box) with the values shown here – click **OK** when done.

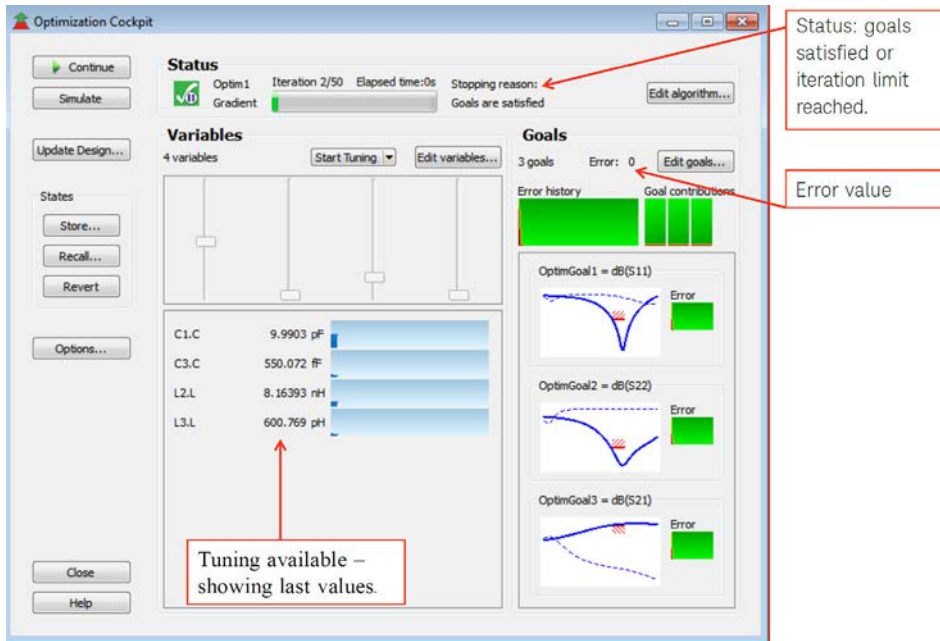
| Simulation Variables Setup | | | | | | | | |
|---------------------------------------|-------------------------------------|-------|------|---------|---------|------|-----|------|
| Tuning Optimization Statistics DOE | | | | | | | | |
| Name | Optimiz | Value | Unit | Format | Min/+/- | Unit | Max | Unit |
| My_Sys_AMP_lib:a2_match_opt:schematic | | | | | | | | |
| M1 | | | | | | | | |
| gw | <input type="checkbox"/> | 200 | um | min/max | | | | |
| _M | <input type="checkbox"/> | 1 | | min/max | | | | |
| L1 | | | | | | | | |
| L | <input type="checkbox"/> | 8 | nH | min/max | | | | |
| R | <input type="checkbox"/> | 1e-12 | | min/max | | | | |
| L2 | | | | | | | | |
| L | <input checked="" type="checkbox"/> | 8 | nH | min/max | 0.5 | nH | 47 | nH |
| R | <input type="checkbox"/> | 1e-12 | | min/max | | | | |
| LFB | | | | | | | | |
| L | <input type="checkbox"/> | 2.7 | nH | min/max | | | | |
| R | <input type="checkbox"/> | 1e-12 | | min/max | | | | |
| C1 | | | | | | | | |
| C | <input checked="" type="checkbox"/> | 10 | pF | min/max | 0.5 | pF | 20 | pF |
| C2 | | | | | | | | |
| C | <input type="checkbox"/> | 10 | pF | min/max | | | | |
| L3 | | | | | | | | |
| L | <input checked="" type="checkbox"/> | 5 | nH | min/max | 0.5 | nH | 47 | nH |
| R | <input type="checkbox"/> | 1e-12 | | min/max | | | | |
| C3 | | | | | | | | |
| C | <input checked="" type="checkbox"/> | 2 | pF | min/max | 0.5 | pF | 20 | pF |

16. Running an Optimization

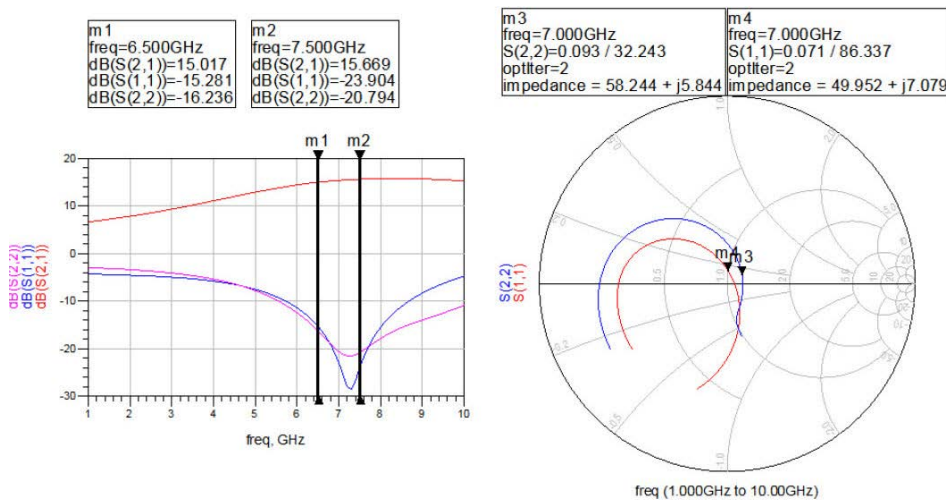
- Click the **Optimize** icon to run the optimization.



- Immediately, you will see the **Optimization Cockpit**, Status window and Data Display appear. The cockpit shows all the information you need.



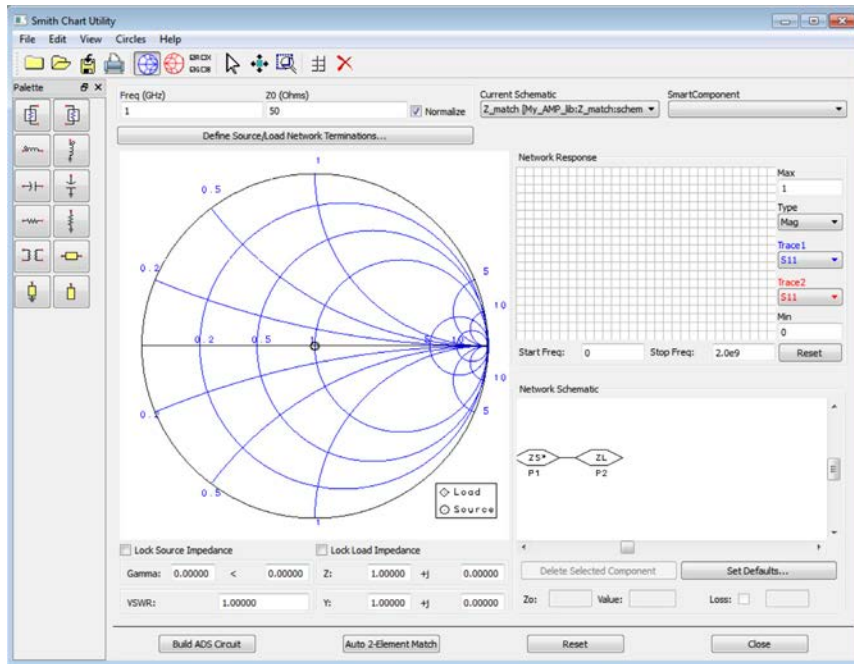
- In the **Data Display** (a2_match_opt) on the rectangular plot, move the **Line Markers** to **6.5 GHz** and **7.5 GHz**. On the Smith chart, set the markers to **7 GHz**. Notice that the all goal were achieved, i.e. $S_{11} < -15$ dB, $S_{22} < -15$ dB and $S_{21} > 15$ dB



- d. Click the **Close** button in the Optimization Cockpit – then click the button: **Don't Update the Design**. This means the L and C values will not be updated – this is OK because you can re-run the optimization to get the results back again.
- e. Save and close the schematic and the data display.

17. Smith Chart Utility for Z Matching Network

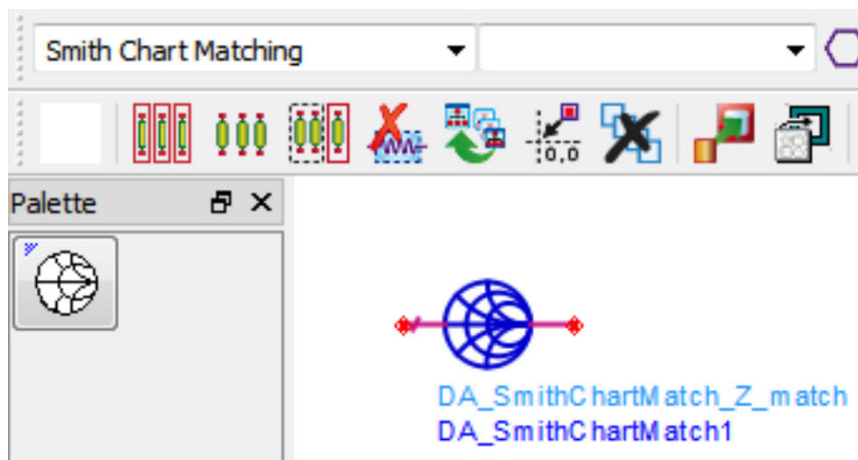
- a. Create a new schematic and name the cell: **Z_match**.
- b. In the new schematic, click on the command: **Tools > Smith Chart ...**



- c. At the top of the Smith Chart Utility window, click the **Palette** icon shown here – this adds the Smith Chart palette with the Smith Chart icon to your schematic.

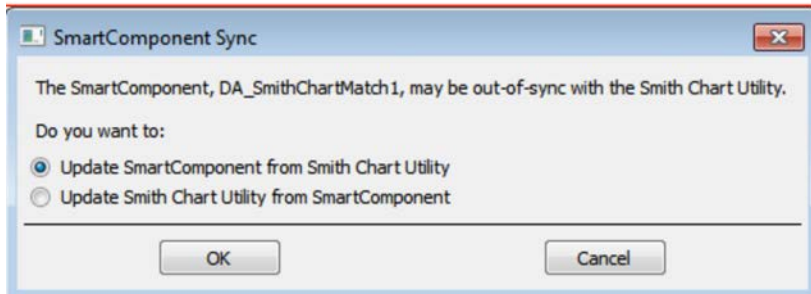


- d. Go back to the schematic window and from the palette, insert the **Smith Chart Matching Network component** (shown here). Click **OK** when a message dialog appears – you can ignore it. It is necessary to insert this component in a schematic for the tool to be used.



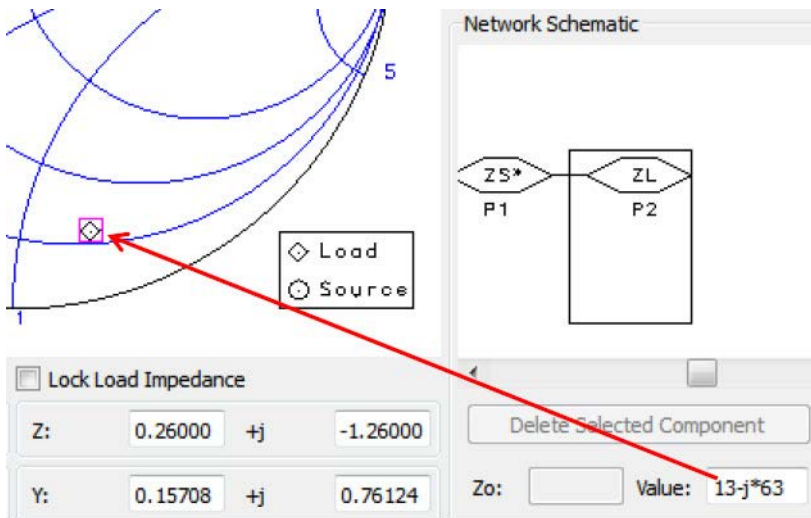
- e. In the Smith Chart Utility (at the top) in the Freq (GHz) field type **7**. When prompted sync the component from the Smith Chart Utility and click OK.

| | |
|------------|-----------|
| Freq (GHz) | Z0 (Ohms) |
| 7 | 50 |



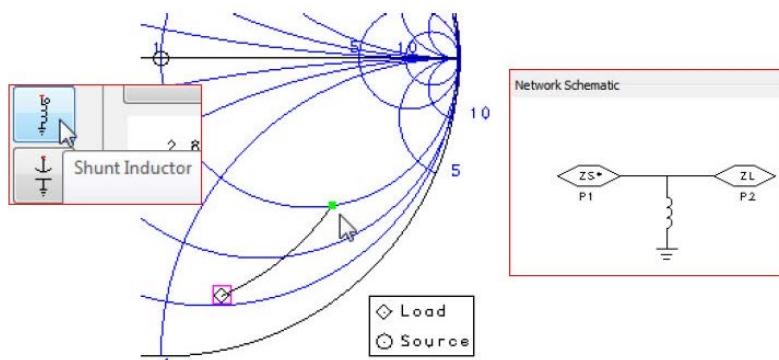
- f. In the lower right corner of the Smith Chart utility window, select the **ZL** component and type in the approximate impedance looking into the amplifier from the last simulation: **13-j*63** as shown here, and press **Enter**.

Notice that the load symbol (diamond) on the Smith chart has relocated to the lower right.

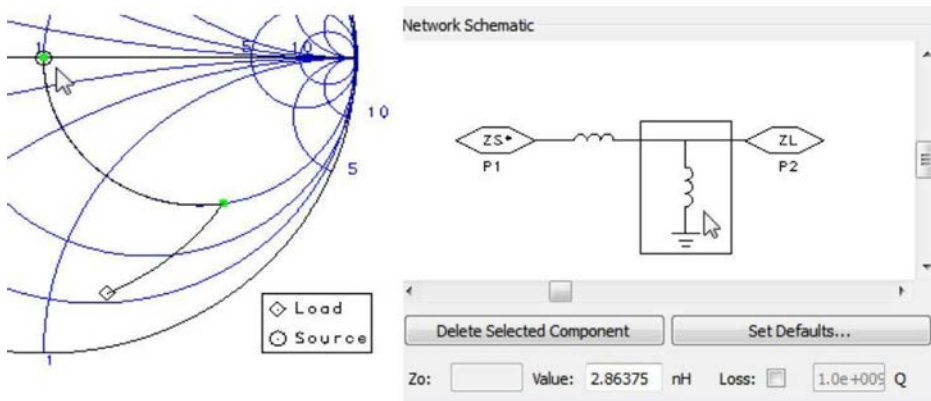


- g. To build the matching network, select the **shunt inductor** from the Smith Chart Utility palette.

Then move the cursor on the Smith chart as shown here – stop when you get to the 50 Ohm circle of constant resistance, **click** to stop; it does not have to be exact.



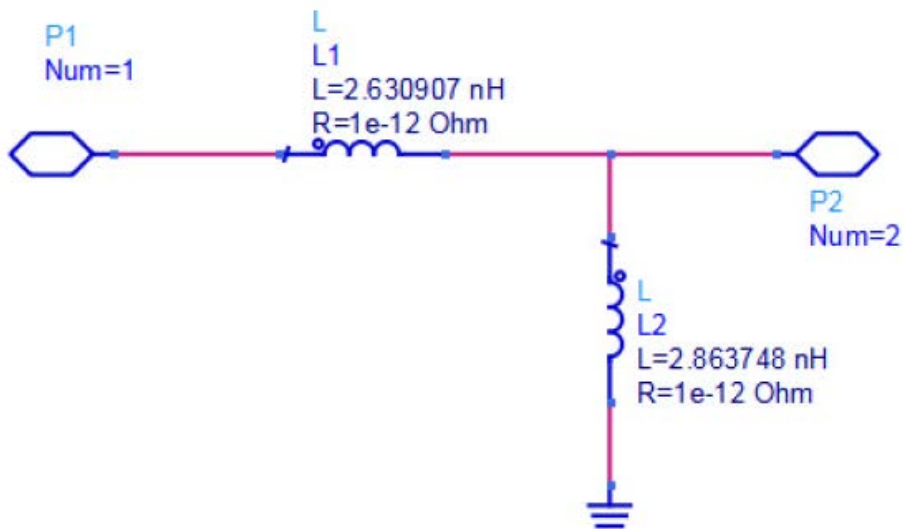
- h. Select the **series inductor** from the palette and then move the marker along the circle to the center of the Smith chart (approximately) and **click**, as shown below.
- i. In the lower right in the Network Schematic pane, select either shunt or series L to see the value, see here.



- j. Click the **Build ADS Circuit** button at the bottom of the Smith Chart Utility window to create the matching. Now you have designed the matching network that was used to obtain the good results in the previous steps. **Close** the utility window.



- k. In the Z_match schematic push into the Smith Chart symbol to see the matching circuit.



Notice the values obtain a very close to the rounded value we used in the matching circuit in the lab. Your values may be slightly different.

- l. You can close or delete the cells or any files used for the Smith chart matching – you don't need them. You only used the utility to design the matching circuit topology.

Congratulations! You have completed Microwave Amplifier Design and Smith Chart Utility for Z Matching Network. Check out more examples at www.keysight.com/eesof-ads-rfmw-examples.

Download your next insight

Keysight software is downloadable expertise. From first simulation through first customer shipment, we deliver the tools your team needs to accelerate from data to information to actionable insight.



- Electronic design automation (EDA) software
- Application software
- Programming environments
- Productivity software

Learn more at
www.keysight.com/find/software

Start with a 30-day free trial.
www.keysight.com/find/free_trials

For more information on Keysight Technologies' products, applications or services, please contact your local Keysight office. The complete list is available at:
www.keysight.com/find/contactus

Americas

| | |
|---------------|------------------|
| Canada | (877) 894 4414 |
| Brazil | 55 11 3351 7010 |
| Mexico | 001 800 254 2440 |
| United States | (800) 829 4444 |

Asia Pacific

| | |
|--------------------|----------------|
| Australia | 1 800 629 485 |
| China | 800 810 0189 |
| Hong Kong | 800 938 693 |
| India | 1 800 11 2626 |
| Japan | 0120 (421) 345 |
| Korea | 080 769 0800 |
| Malaysia | 1 800 888 848 |
| Singapore | 1 800 375 8100 |
| Taiwan | 0800 047 866 |
| Other AP Countries | (65) 6375 8100 |

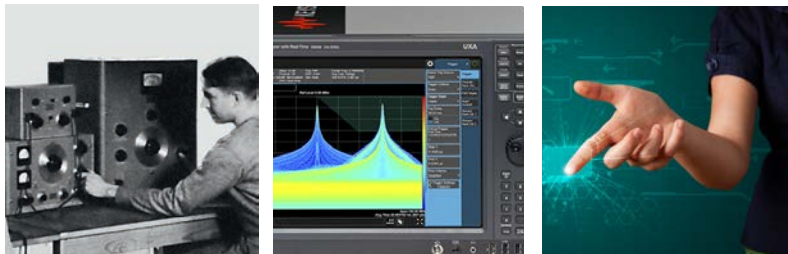
Europe & Middle East

| | |
|----------------|---------------|
| Austria | 0800 001122 |
| Belgium | 0800 58580 |
| Finland | 0800 523252 |
| France | 0805 980333 |
| Germany | 0800 6270999 |
| Ireland | 1800 832700 |
| Israel | 1 809 343051 |
| Italy | 800 599100 |
| Luxembourg | +32 800 58580 |
| Netherlands | 0800 0233200 |
| Russia | 8800 5009286 |
| Spain | 800 000154 |
| Sweden | 0200 882255 |
| Switzerland | 0800 805353 |
| | Opt. 1 (DE) |
| | Opt. 2 (FR) |
| | Opt. 3 (IT) |
| United Kingdom | 0800 0260637 |

For other unlisted countries:
www.keysight.com/find/contactus
(BP-06-08-16)

Evolving

Our unique combination of hardware, software, support, and people can help you reach your next breakthrough. **We are unlocking the future of technology.**



From Hewlett-Packard to Agilent to Keysight



myKeysight

myKeysight
www.keysight.com/find/mykeysight
A personalized view into the information most relevant to you.